FlowVision FlowVision FlowVision

FlowVision

Version 3.09.04

User's guide

© CAPVIDIA, 1999-2015 Leuven, Belgium. All rights reserved.

Contents

1 Quick start 2
1.1 Example of a problem (mixing a liquid) 3
1.2 Loading a geometric model of a mixer 4
1.3 Operations in the View window 7
1.4 Defining general settings of the project 10
1.5 Defining Substances and their parameters 12
1.6 Defining Phases and their parameters 15
1.7 Defining a Model and its parameters 17
1.8 Indicating a Model in calculation subregion 18
1.9 Defining Boundary conditions 19
1.10 Defining Initial conditions 24
1.11 Defining a computational grid 25
1.12 Defining simulation controls 30
1.13 Displaying results 33
1.13.1 Creating charachteristics 36
1.13.2 Creating layers 38
1.14 Carrying out the project calculations 42
1.15 Monitoring the project calculation process 43
1.16 Creating an animation 46

© CAPVIDIA, 1999-2015 Leuven, Belgium. All rights reserved.

1 Quick start

This section shows a case study of a full cycle of the project - from the formation of the project to analyze the results.

After stepping through the actions described here, you will gain basic skills of working with the interface FlowVision.

As an example, consider task fluid mixing in a mixer.

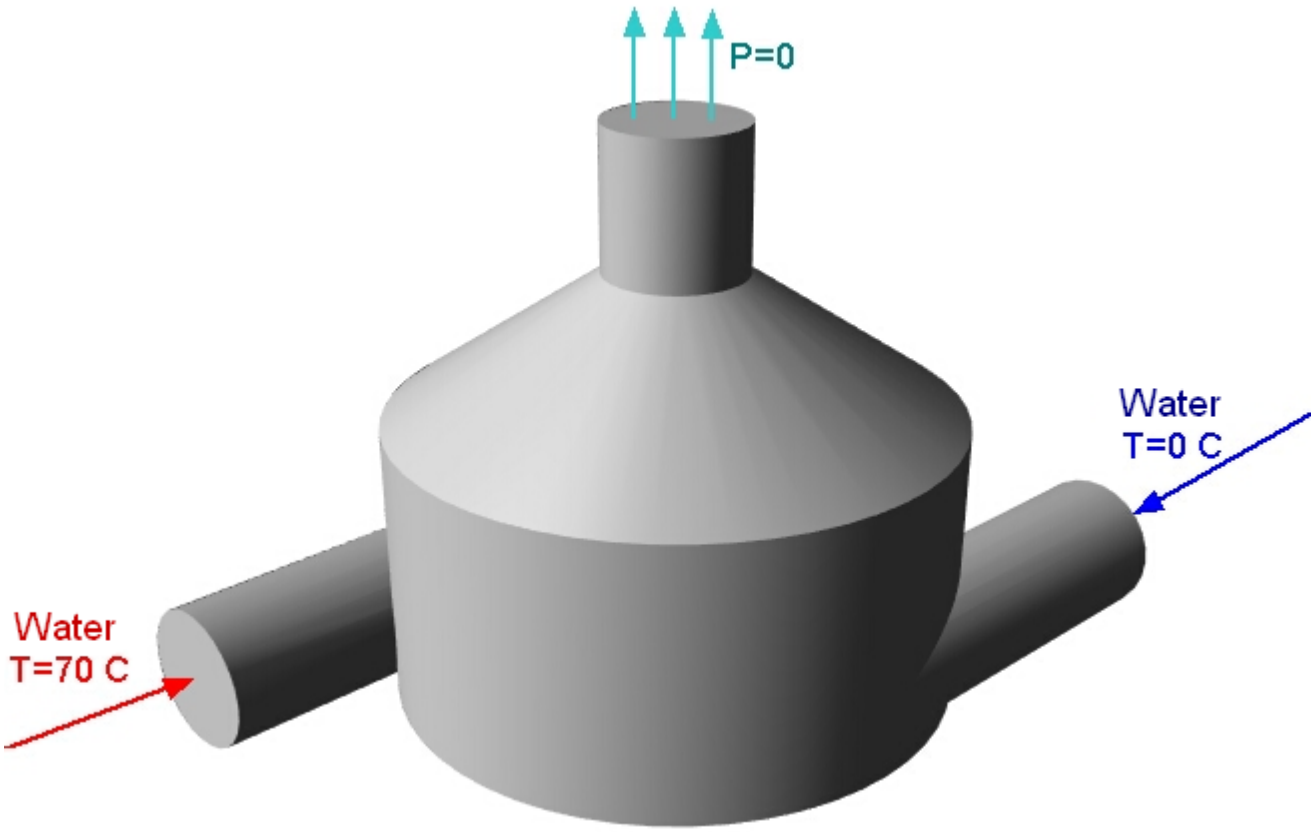
Sequence of actions:

1. Download the geometry
2. Discover basic operations in the View
3. Specify the general settings of the project
4. Ask **Substances** (In this example - one substance **Water**)
5. Ask **Phase** (Liquid water), and its parameters
6. Ask **Model** and its parameters
7. Specify **Model** in computational subregion
8. Ask **Boundary conditions** tap
9. Explore the concept **Initial conditions**
10. Ask a computational grid (ie, set **Starting grid** and configure **Adaptation**)
11. Set the control parameters calculation
1. Adjust the display of the results:
    a. establishment characteristics
    b. creation of layers
12. Practice with the procedures start, stop and continue the calculation
13. Adjust the control to calculate project
14. Create a series of graphic files Trending of the calculation time (Animation)

© CAPVIDIA, 1999-2015 Leuven, Belgium. All rights reserved.

1.1 Example of a problem (mixing a liquid)

Consider the flow of water in the mixer, which is a tank with two supply tube at the base and one outlet tube on the cover:



Tank diameter is 0.04 m, the height of the cylindrical part of the tank - 0.02 m, height of the cover - 0.01 m, diameter of the tubes - 0.01 m. Through one of the tubes served cold water (T = 0^{o}C), through the other - hot (T = 70^{o}C) the water flow in the supply tube is the same and equal to 0.1 kg $^{-1}$.

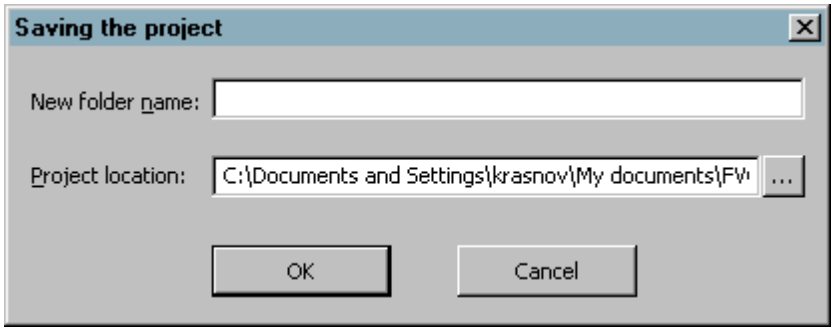
The purpose of the calculation is to obtain the flow pattern and temperature equalization of water in a blender.

© CAPVIDIA, 1999-2015 Leuven, Belgium. All rights reserved.

1.2 Loading a geometric model of a mixer

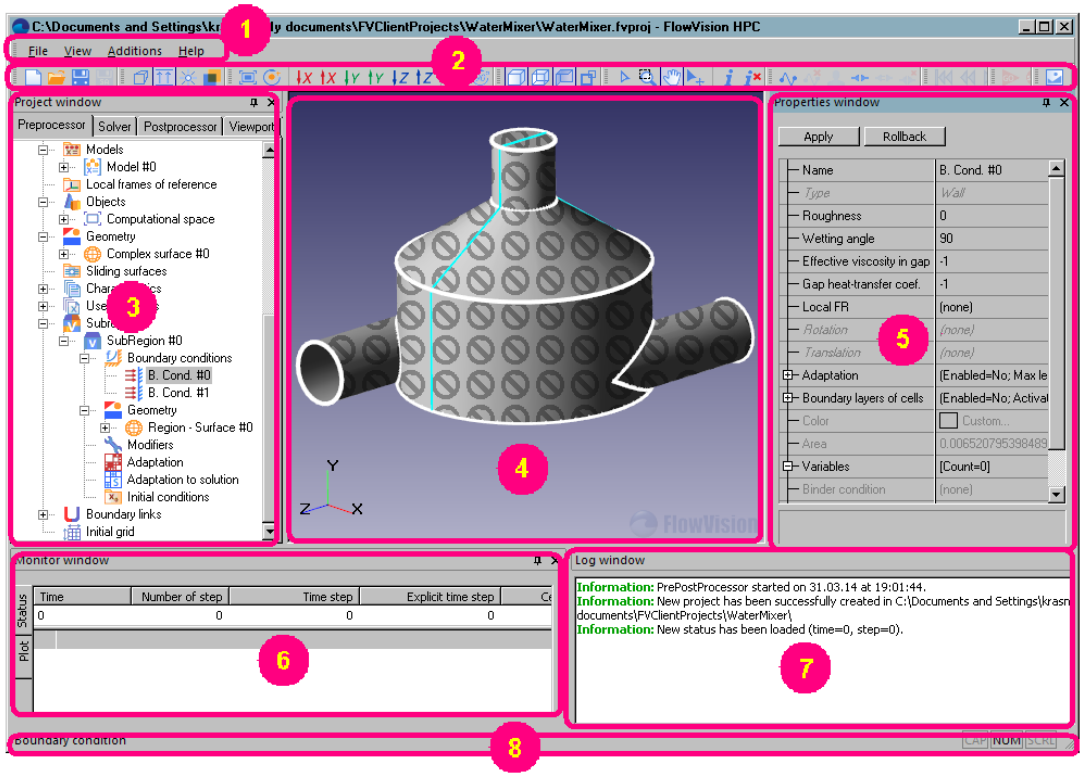
The geometry of the computational domain, on the basis of which the project is created, create a software package is*FlowVision*in geometric modeling systems.

Formation of the project begins by loading the geometric model of the computational domain. Perform the following steps:

Step	Actions
1	Start**Pre-Postprocessor**(Start Button Windows> **Programs> FlowVision>Pre-Postprocessor).** Window of **Pre-Postprocessor** will open. Before the launch of**pre-postprocessor**to be launched **License Manager**. Start of**License Manager**can be tuned so it can occur automaticallyat system startup. If necessary,**License Manager**can be started manually via the**Start**buttonWindows **(Start>Programs> FlowVisionLM>License** Manager),or run it executable **(`FvLicense.exe`**Windows or**`FvLicense`**in Linux).
2	Select main menu Choose **File>New. The standard window for selecting a file.**
3	Select the file**`Mixer.wrl, located in the folder:`** • **`C: \ Program Files \ TESIS \ FlowVision HPC \ Tutorial \ Samples \ Geom`** • *or* **`C: \ Program Files (x86) \ TESIS \ FlowVision HPC \ Tutorial \ Samples \ Geom`**
4	Dialog box to specify the name and location of the project:  Specify the name of the project in it in the**name of the new folder**(and, if necessary, the location in the**Location**field), then click**OK.** In the client part of the project will create a subdirectory of the project that contains the client files of the project. In the Project Tree tab **Preprocessor** appear loaded elements of the project. After you create a project in the tree**Preprocessor**error messages are displayed in the geometry, marked with **"!":** • load geometry to another **Tolerance** • in the context menu **Region** click**Fix self-intersection** • Use to correct errors in the geometry of a specialized package (for example, 3DTransVisia) Loading a previously created and saved project**FlowVision**via **File>Open, where in the dialog box select the appropriate project.** If you can not download the file geometry, then: • checkwhether he FlowVision supported format, and its elements are the requirements • send thefileto thegeometry technical support FlowVision with a description of the problems encountered.

© CAPVIDIA, 1999-2015 Leuven, Belgium. All rights reserved.

5	In the review (4)**pre-postprocessor**appears loaded geometric model of the computational domain.



Window pre-postprocessor

In the illustration, the numbers are the components of the windows**pre-postprocessor:**

- the main menu (1)
- toolbars (2)
- **Project**window(3)
- Window**View**(4)
- **Properties**window(5)
- **Protocol**window(6)
- **Monitoring**window(7)
- status bar (8)

In the project window(3)the project tree is displayed in four tabs: **Preprocessor,Solver,Postprocessor,Viewport.**

After creating the project and download the geometrical model of the computational domain, all folders of the project tree empty folders except the geometric model of the computational domain and its display parameters:

- **Preprocessor**tabgeometric model of the computational domain is displayed in the folder**Region>Subregions >subregion # 0>Geometry, automatically generated when loading the boundary conditions are located atRegion>Subregions >subregion # 0>Boundary** conditions;
- **Postprocessor**tabdisplay options boundary conditions are placed in folders**3D-scene> Objects> Computational space > Solids > Subregions > SubRegion # 0> Boundary conditions** and **3D-scene > Layers > Solids > Subregions > SubRegion # 0> Boundary** conditions.

In addition to the geometric model of the computational domain in the project tab**Postprocessor**automatically creates a**plane # 0**in the folder**3D-scene> Objects> Computational** space. The plane is displayed in the**View**(4) the vertical line of blue color. **Plane # 0**is oriented perpendicular to the X axis and passes through the center of the computational domain. This plane is not part of the geometric model of the computational domain, so it remains invisible to the current in the mixer of the liquid (ie it - completely transparent to the liquid and has no influence on it). The plane is necessary in order to display it on the calculated distribution of the values computed characteristics and perform other auxiliary operations.

© CAPVIDIA, 1999-2015 Leuven, Belgium. All rights reserved.

© CAPVIDIA, 1999-2015 Leuven, Belgium. All rights reserved.

1.3 Operations in the View window

Before you generate the project, consider loaded geometric model of the computational domain.

Additional object**plane # 0**in the folder **3D-scene>Objects>Computational space**allows to cut part of the geometric model. Operations in the**View**window performed using the toolbar buttons and mouse.

Image of a geometric model of the computational domain

To adjust the image:

Step	Actions
1	Click the button to turn off the effect of perspective and be sure to click enough (to fill the surface geometric model).
2	Click to display facets, making up the framework of the model surface. Press this button.
3	Click for mode conversion type.

To rotate an image:

Step	Actions
1	Click the button to display the internal and external surfaces of the object
2	Click , , , , , get the views of the mixer along the axes of the coordinate system associated with the model of the computational domain - absolute coordinate system. The contour of the cross section of the mixer object **plane # 0** is displayed in blue.
3	In order to rotate the image of the mixer in the**View:** a) move the mouse pointer in the**View**window b) click the left mouse button and keep it pressed, move the mouse pointer For a reference, use the icon of the coordinate system, showing the unit vectors of the coordinate system (in its default configuration it is located in the lower left corner of the**View.**
4	Click to hide the external surfaces of the object and turn the faucet in the desired manner (repeat steps**3,4).**

To offset and scale the image:

Step	Actions
1	In order to zoom in the mixer window**View:** a) move the mouse pointer in the**View**window b) rotate the mouse wheel. Image is increased or decreased relative to the center of the window**View.**
2	In order to shift the image in the mixer**View** a) move the mouse pointer in the**View**window b) click the right mouse button and keep it pressed, move the mouse pointer.
3	To return to the original state of the image in the**View**windowand display the entire geometric model of the computational domain, click.

© CAPVIDIA, 1999-2015 Leuven, Belgium. All rights reserved.

To display groups of facets that make up the geometric model:

The geometric model of the computational domain consists of several groups of facets (to display borders groups, click).

Step	Actions
1	Press to activate the selection.
2	To display one of the groups of facets: a) move the mouse pointer on the geometric model and click the left mouse button; on the screen will be only one of the groups of facets lying beneath the point of clicks. One side of the group is a front surface and painted (by holding down the). On the reverse side of the surface pattern applied to the group of fragments ; b) after re-clicking the left mouse button on the screen will display the next group of facets, which lies under the point-click (if available).
3	To display the entire geometric model, set the mouse pointer in the **View** window is a geometric model and click the left mouse button.

Cross-section of a geometric model of the plane

In order to cut off a plane geometric model of the computational domain:

Step	Actions
1	Perform the following two conditions: • plane should be cutting, which **Postprocessor** tab right-click the line **3D-scene>Objects>Plane # 0** and set the mark in the paragraph **to the** View; • geometric model of the computational domain must be cut-off, which **Postprocessor** tab right-click the line **3D-scene> Objects> Computational space> Solids** and check the box in paragraph **Apply clipping**.
2	Highlight tab **Postprocessor** line **3D-scene>Objects>Plane #** 0. In **the View** window visible plane, the reference point on it, and coming from the reference point of the vector normal to the plane. It can be seen that the plane cuts the negative half-space (the normal vector in the positive half-watching).

To change the position of the plane with the mouse:

Step	Actions
1	Press to select **Edit** object.
2	Highlight tab **Postprocessor** line **3D-scene>Objects>Plane #** 0.
3	To rotate the plane around the reference point, click the left mouse button and keep it pressed, move the mouse pointer.
4	To move the reference point along the plane right-click and hold, and drag the mouse.
5	To move the reference point along the normal to the plane, press the left and right mouse button and hold it, move the mouse pointer.

© CAPVIDIA, 1999-2015 Leuven, Belgium. All rights reserved.

Changing the position of the plane with the help of the window "Properties":

Step	Actions
1	Highlight tab**Postprocessor**line**3D-scene>Objects>Plane #** 0.
2	In the**Properties**window, follow these steps: • for the orientation of the plane along the axis X (Y, Z), click**Object> Operations>** And then click**Apply;** • to change the orientation of the plane on the opposite, click**Object> Operations>** and then click**Apply.**

© CAPVIDIA, 1999-2015 Leuven, Belgium. All rights reserved.

1.4 Defining general settings of the project

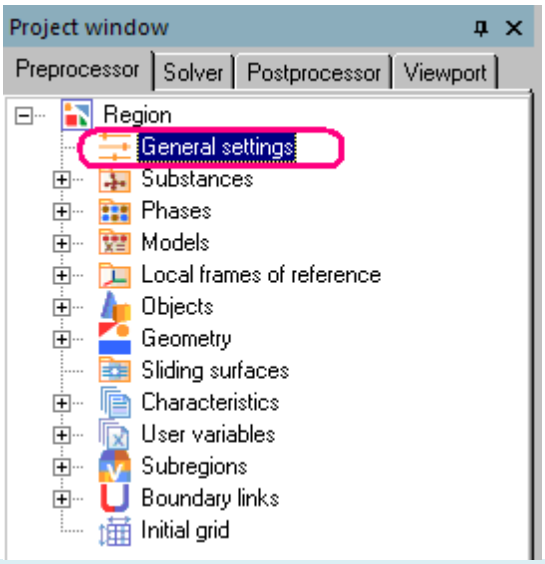
Some of the common parameters include values such as:

- reference **temperature** T_{ref} and **pressure** P_{ref} (default is 0^oC and 101000 Pa)
- vector *of gravitational acceleration* in the absolute coordinate system associated with the geometric model of the computational domain **(gravityvector).** In the example in the figure given the acceleration of gravity 9.8 m / s^2 in the direction *against the* axis Y.
- **g-point** - the coordinates (in the absolute coordinate system) the point at which a hydrostatic pressure component vanishes
- **g-density -density** of a heavy fluid, situated above the g-point (as if the layers of a heavy fluid is not specified, then the lower g-point), whichcontributes to the hydrostatic component of the pressure

Some of the general guidelines of the project also includes the definition of the layers of a heavy fluid, below the g-points, each of which is characterized by the following parameters:

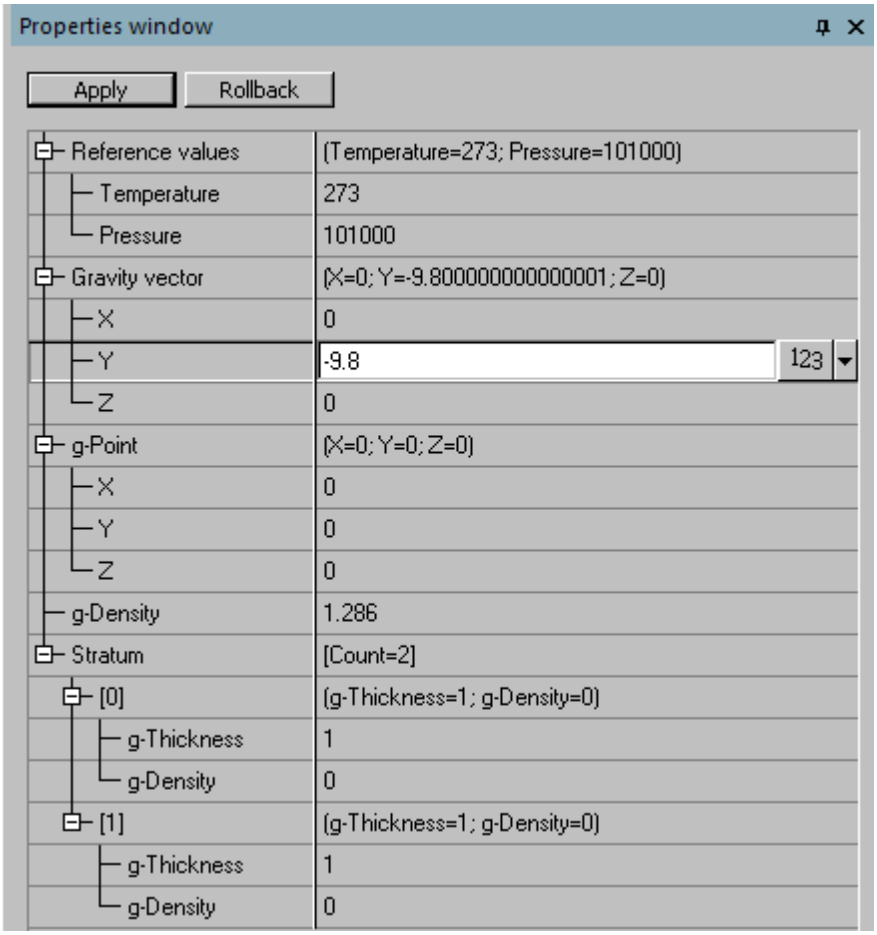
- **g-thickness** - the thickness of a heavy fluid
- **g-** density-**density** of a heavy fluid in a given layer (lower layer and if the next layer is not defined)

The **General settings** are specified in the **Properties** of the element **Region > General settings** in the project tree (in the tab **Preprocessor**):



Element "General settings" in the project tree

© CAPVIDIA, 1999-2015 Leuven, Belgium. All rights reserved.



Example of a common project settings

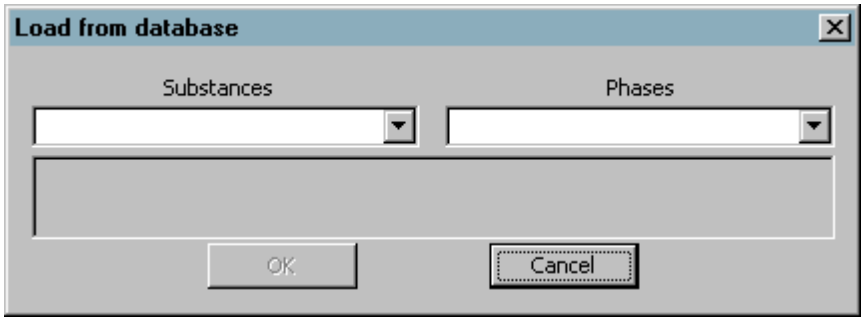
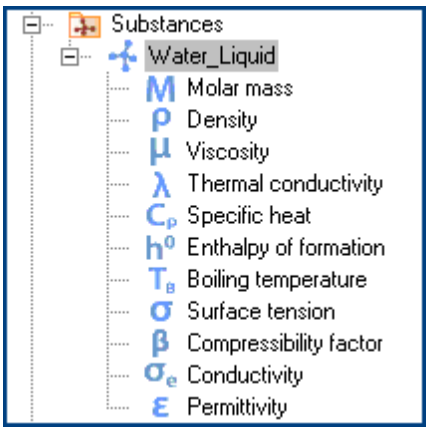
© CAPVIDIA, 1999-2015 Leuven, Belgium. All rights reserved.

1.5 Defining Substances and their parameters

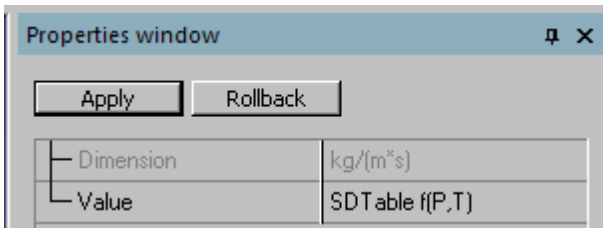
*Substance*is determined by the state of aggregation and properties.
Substancesare createdin the foldertree**Preprocessor Substances**. Aggregative state of a**Substance**is defined in the**Substance**'s **Properties**window. Each**Substance**has a set of child elements corresponding to physical properties of the substance. Values of thesephysical propertiesofthe substance are defined in the**Properties**windowsofappropriate child elements. Parameters of a**Substance** in the projectcan be defined manually or loaded fromthe**Substance** Database. The number of substances may be one or more.

We define the physical properties of the current environment, which in this project is a single-phase and consisting of one substance - water, considered as an incompressible fluid.

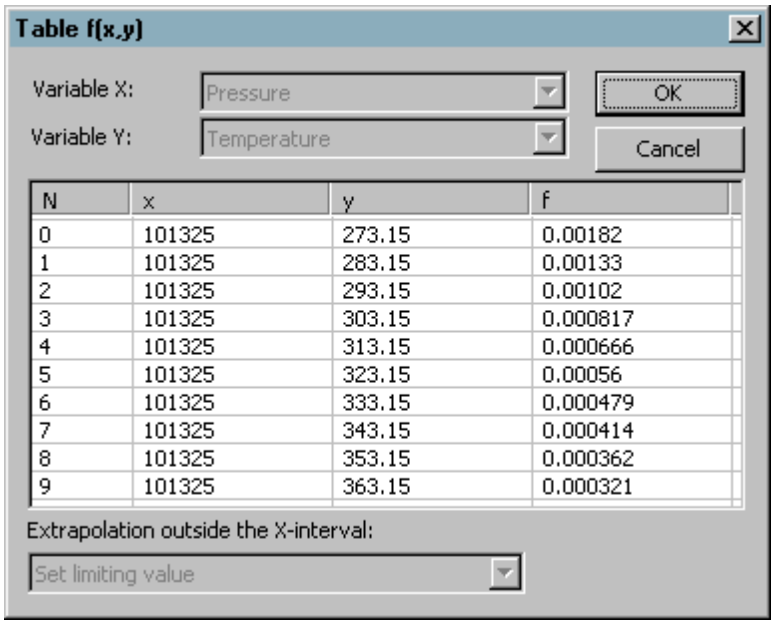
Setting substance

Step	Actions
1	Right-click the folder row**Substances**Preprocessortab and choose**Create.** In the folder**Substances**folder is displayed new element**Substance #** 0.
2	Right-click the folder row**Substance #** 0,and then click**Load from SD > Standard.** **Download**dialog box is displayed**from the** database: 
3	Select**Water**in the drop-down listand the value of**Liquid**Substancesin the drop-down list **Phases, then click**OK. In the folder**Substances**shows the new element**Water_Liquid.**
4	Expand element**Water_Liquid.** Displays a list of physical properties of matter: 
5	To view the specified properties (for example,**viscosity):** a) scroll to the**viscosity of**the list of the physical properties of the substance; in the**Properties**windowdisplays **theviscosity**parameter:

© CAPVIDIA, 1999-2015 Leuven, Belgium. All rights reserved.



b) In the**Properties**window, click on the**Value**field(on the field itself for data entry), and then on the icon . **Table**dialog box is displayed**f (x)**with a tabulated function of the viscosity of the pressure (X) and temperature (Y):



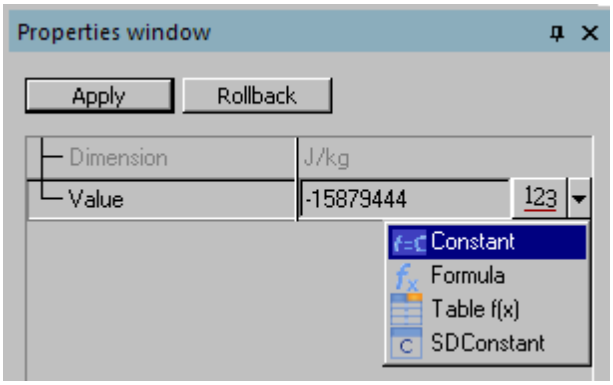
N	x	y	f
0	101325	273.15	0.00182
1	101325	283.15	0.00133
2	101325	293.15	0.00102
3	101325	303.15	0.000817
4	101325	313.15	0.000666
5	101325	323.15	0.00056
6	101325	333.15	0.000479
7	101325	343.15	0.000414
8	101325	353.15	0.000362
9	101325	363.15	0.000321

c) close the**table.**

6 Specify a value of zero **Enthalpy of formation**(this option is only used in the model of mass transfer with chemical reactions or burning)

a) highlight **enthalpy of formation of** the list of physical properties of the substance

b) In the **Properties** window, click the mouse pointer on the row **value** and then click on :



c) drop-down list, select **Constant**

© CAPVIDIA, 1999-2015 Leuven, Belgium. All rights reserved.

d) in the input box set the value to **0**, and then click **Apply**:

For reference: physical properties of water

Physical property	The value of	The dimension of
Density	1000	[Kg / m^{3}]
Viscosity	0.001	[Kg / (mη)]
Thermal conductivity	0.6	[W / (m K)]
Specific heat	4217	[J / (kg K)]

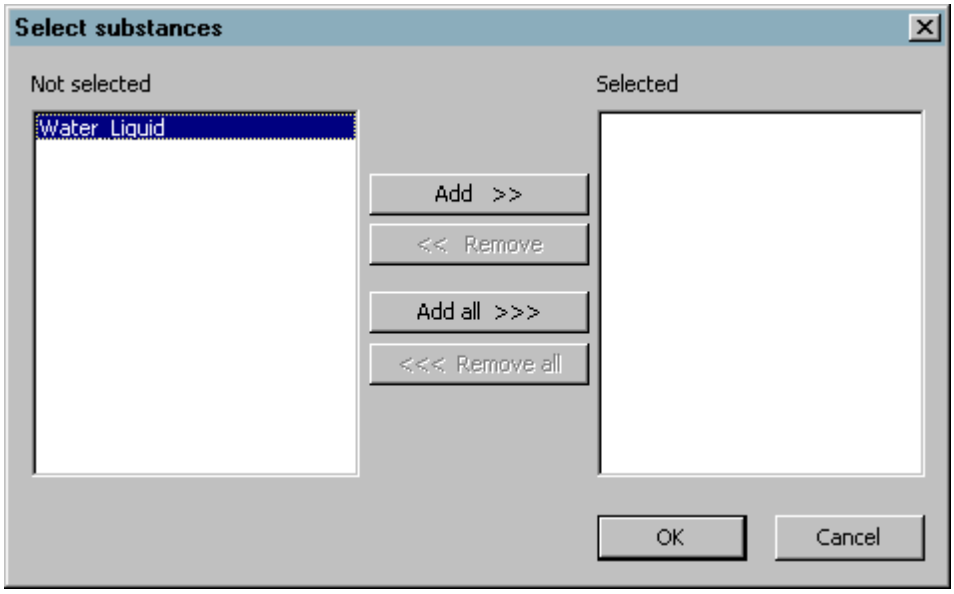
© CAPVIDIA, 1999-2015 Leuven, Belgium. All rights reserved.

1.6 Defining Phases and their parameters

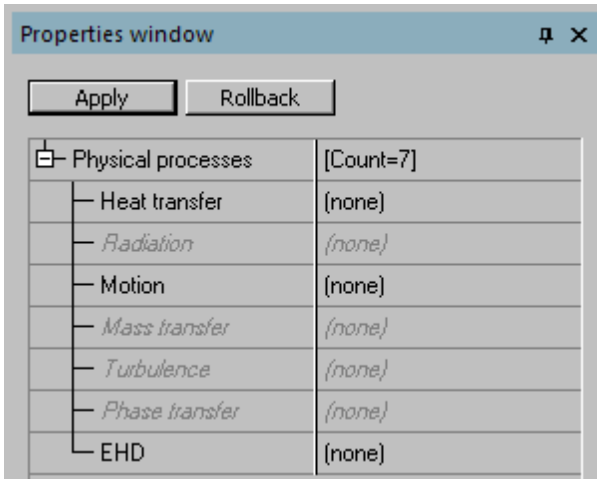
Phase- a set of agents and the simulated physical processes.
Phases are defined in the**Preprocessor**treeinfolder **Phases**. The project should be at least one phase. Number of phases is not limited.

We indicate what substance is phase and define the physical processes in the phase of which will be considered in the calculation.

Setting phases:

Step	Actions
1	Right-click the folder **Phases** and select the **Create continuous**. In yourfolder is displayed **Phases** new element**Phase #** 0.
2	Expand the created element, right-click the lineand select thefolder**Substances**in it**Add /** remove. Dialog box, click**the** substance: 
3	Select the desired substance in the list on the left**unselected**(in our case -**Voda_Zhidkaya) and clickAdd.** The selected material is displayed in the right list**selected.**
4	Click**OK:** In the folderis displayed**Substances**added element.
5	Select the item,**Physical processes**in the folder**Substances.** In the**Properties**windowdisplays a list of physical processes.

© CAPVIDIA, 1999-2015 Leuven, Belgium. All rights reserved.



6

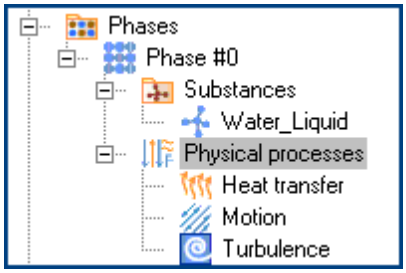
Select the drop-down lists the physical processes in the following order:

- parameter **Heat transfer** select from the list of value **Convection & conduction**
- parameter**Movement Motion**select from the list the value of**Newtonian fluid**
- **Turbulence**parameter value, select from the list of**KES**(standard k-ε turbulence model). Please note thatwhile the**movementMotion= (none) (None)**turbulence model can not be set)

7

Click**Apply.**

Specified items phase displayedin the project tree:



© CAPVIDIA, 1999-2015 Leuven, Belgium. All rights reserved.

1.7 Defining a Model and its parameters

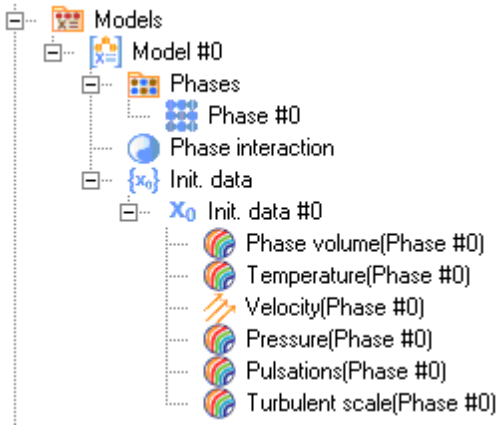
Model- a set of phases and phase interactions.

Models are createdin the foldertree**Preprocessormodels. The project should be at least one model. Number of models is not limited.**

In the presence ofseveral**phases**in the**models**can be specified**interfacial** interaction.

At this stage, the formation of the project will determine which phase is included in the model, and define the initial data, which later will be given the initial conditions for the model equations.

Setting Model

Step	Actions
1	Open the context menu of the folder**models**and select the**Create**command. In the folder**Models**folder is displayed new element**Model #** 0.
2	Expand**Model #** 0,right-click the lineand select thefolder**Phases**it**Add /** remove. Dialog box, click**the** phase.
3	Select the desired phase in the left list**Undrawn**(in our case -**Phase #** 0),and then click**Add.** The selected phase is displayed in the right list**selected.**Click**OK.**
4	In the folderis displayed**Phases**added element. **Initial** folder. data> Setup. data **# 0**displays a list of items***Initial value of*** variable: 
5	If you highlightin the project treeelement***Initial value of*** variable,then in the**Properties**windowdisplays the value of the initial data, which by default is zero. This means that initially the liquid in the tank is at rest and at a temperature equal to the reference value, i.e., 0 degrees Celsius.

For in this example (mixer)

Ask a**Model:**

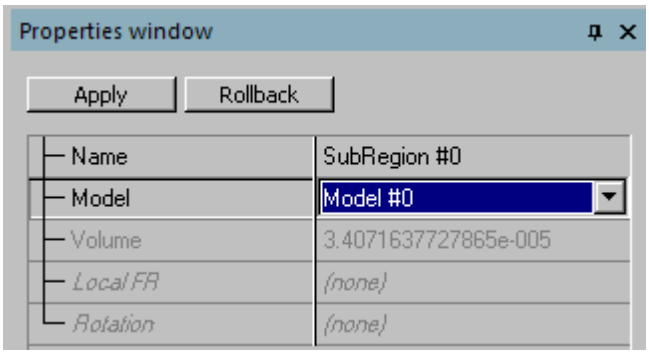
- In the context menu,select**Create a**folder**Models**
- In**Phases**in the**Models**folder**# 0# 0**add**Phase**
  - In the context menu of a folder**Model # 0>Phases, click****Add / Remove**
  - In the form, select otryvsheysya**Phase #** 0,and then click**Add**

© CAPVIDIA, 1999-2015 Leuven, Belgium. All rights reserved.

1.8 Indicating a Model in calculation subregion

We indicate the model estimated subregions (ie, subregions, which will be calculated), not doing it for the off-design subregions.

To specify the model in the computational subregion:

Step	Actions
1	Select the folder in the project tree **Subregions > Subregion #0**.
2	In its **Properties** window, set the parameters **of** the **Model** by selecting from the drop-down list, select **Model #0**: 
3	Click **Apply**.

© CAPVIDIA, 1999-2015 Leuven, Belgium. All rights reserved.

1.9 Defining Boundary conditions

Boundary condition- the condition imposed on the design variableson borders of a**Subregion.Boundary conditions**are specified in the folder **Boundary conditions** in an appropriate**Subregion.**

> Before you set the**boundary conditions**in the folder properties**subregion**must select a predefined**model.**

When loading a geometric model of the computational domain model surface is divided into groups of facets**Region- Group** #N,listed on the**Preprocessor**tabin the folder **Subregions > Subregion #0 > Geometry > Region- Surface # 0** (see. illustration).

> If you use the default settings elements **Region- Group #N** do not appear in the project tree (folders **Subregions > subregion #N> Geometry>Region - Surface #N**),because in real problems it can lead to tens or hundreds of elements **Region- Group** #N).
>
> To display the **Region- Group #N** set in the basic settings **Display > Show All Groups =** Yes.

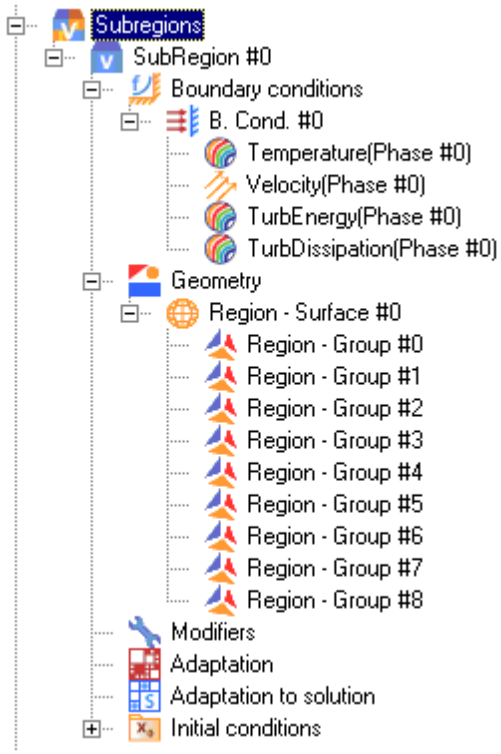


Fig. 3.2. Folder «Subregions»in the project tree

In order to place the boundary conditions on these groups of facets, you must first create them in a folder**Subregions > subregion # 0>Boundary** conditions. When loading a geometric model of the computational domain in this folder are automatically created only one boundary condition**B. Cond # 0**(thermally insulated**Wall).**

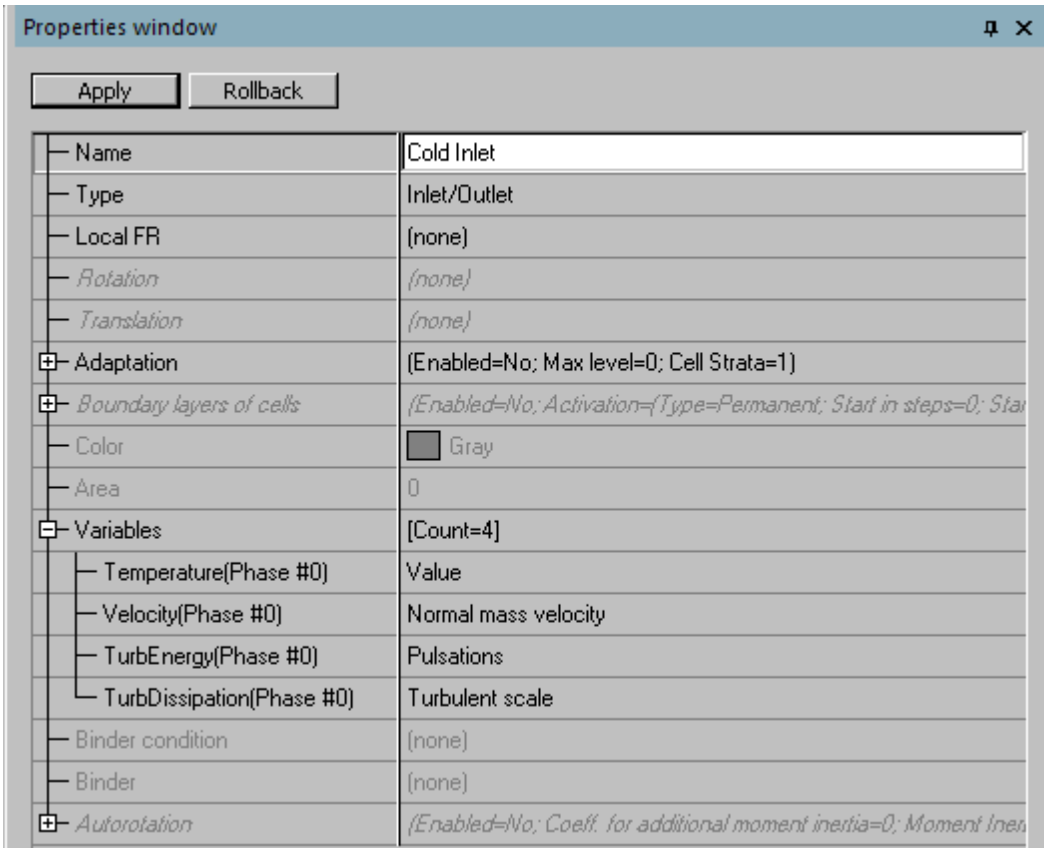
By the boundary conditions are placed in groups more facets.

After placement of the boundary conditions and finding the surface area of each boundary condition are given by the value of the boundary conditions.

Create boundary conditions

Step	Actions
1	Scroll to item**B. Cond # 0**in the folder**Subregions > subregion # 0>Boundary** conditions,in the**Properties**window,set the valuefor the parameter**Wall's name**and click **Apply.**

© CAPVIDIA, 1999-2015 Leuven, Belgium. All rights reserved.

2	Open the context menu of a folder**Subregions > subregion # 0>Boundary conditions**and select the**Create**command. In the folder is added to item**B. Cond #** 1.
3	Scroll to item**B. Cond # 1**and in the**Properties**window: • select**Inlet / Outlet****Type setting** • specify a valuefor the parameter**Cold inlet****Name** • click **Apply:** 
4	Make two copies of the boundary condition**Login** cold,opening the context menu on the element**Login cold**and selecting **the****Copy.** In the folder are added **elements includecold # 0**and**# 1 Login** cold.
5	Select the item**Login cold # 0**in the**Properties**window: • specify a valuefor the parameter**includes a hotname** • Click**Apply**
6	Select the item**Login cold # 1**in the**Properties**window: • **Free**select**outputtype**setting • set the**output**valuefor the parameter**name** • Click**Apply**

The alignment of the boundary conditions on the groups of facets

The alignment of the boundary conditions on the groups of facets produced enumeration groups and indicating for each of them the desired boundary condition. Because all groups of facets is already a boundary condition**Wall, remains only to change the boundary conditions at the inlet and outlet sections of the tubes.**

Step	Actions

© CAPVIDIA, 1999-2015 Leuven, Belgium. All rights reserved.

1	If the itemsin the folder**Subregions > subregion # 0>Geometry>Region- Surface # 0>Region- Group #N**do not appear in the project tree, adjust their display in basic settings FlowVision (Open the menu command **File> Preferences**,setting the **Display > Show All Groups = Yes**).
2	Select the item**Subregions > subregion # 0>Geometry>Region- Surface # 0>Region- Group #1, and in theProperties**window, select**Output**parameter**boundary** condition,and then click **Apply.**
3	Select the item**Subregions > subregion # 0>Geometry>Region- Surface # 0>Region- Group #6, and in theProperties**window, select**Login cold**parameter**boundary** condition,and then click **Apply.**
4	Select the item**Subregions > subregion # 0>Geometry>Region- Surface # 0>Region- Group #8, and in theProperties**window, select**Log hotboundary condition**for the parameter,and then click **Apply.**

Specifying the values of the boundary conditions

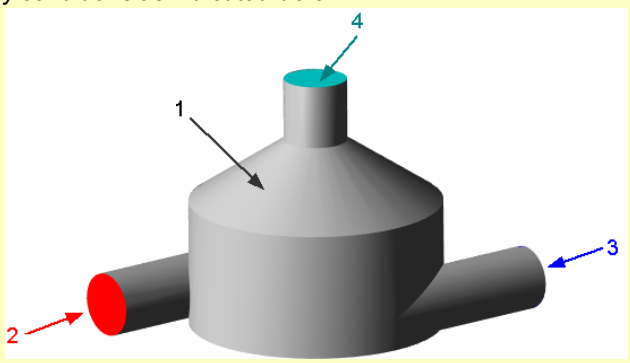
Step	Actions
1	In the folder**Subregions > subregion # 0>Boundary** conditions,highlight the**Log** cold,in the**Properties**window,find the sectional area of the supply tube - 7,7777591212273E-05 $m^{2.}$
2	In the folder**Subregions > subregion # 0>Boundary conditions>Login** cold: a) scroll to the**Velocity,** in the**Properties**window,set the valuefor the parameter**1285.7mass** velocity,and then click **Apply; 1285.7 value obtained by dividing the flow rate of 0.1 kg** $s^{-1}$by the area of the supply tube 7,7777591212273E-05 $m^{2;}$ b) scroll to the**temperature,** In the**Properties**window, specify a value of 0 for the parameter**value, and then click Apply.**
3	In the folder**Subregions > subregion # 0>Boundary conditions>Sign** hot: a) scroll to **Velocity,**in the**Properties**window, set the valuefor the parameter**1285.7mass** velocity,and then click **Apply;** b) scroll to the **temperature,** in the **Properties** window,set the value of **70** for the parameter **value**, and then click **Apply.**

© CAPVIDIA, 1999-2015 Leuven, Belgium. All rights reserved.

For in this example (mixer)

Ask four boundary conditions: ***Wall, Entrance for hot*** *water,* ***cold water entrance*** *and* ***Gate:***

- *Define* ***Model # 0*** *in* ***SubRegion # 0***
- ***Boundary conditions*** *in the folder through the context menu, create three boundary conditions (one already created by default)*
- *Arrange the boundary conditions as indicated below:*



- *Set the parameters* ***of the boundary*** *conditions:*
  - ***Boundary 1***

Type			***=Wall***	
Variables				
	Velocity		***= Logarithm law***	
	Temperature		***= Zero flow***	
	TurbEnergy		***= Value in cell near wall***	
	TurbDissipation		***= Value in cell near wall***	

  - ***Boundary 2***

Type			***=Inlet / Outlet***	
Variables				
	Velocity		***Normal = mass velocity***	
		Value	***=1285.7***	*[Kg m^{-2}s^{-1}]*
	Temperature		***= Value***	
		Value	***= 70***	*[C]*
	TurbEnergy		***= Pulsations***	
		Value	***= 0***	
	TurbDissipation		***= Scale of turbulence***	
		Value	***= 0***	*[m]*

  - ***Boundary 3***

Type			***=Inlet / Outlet***	
Variables				
	Velocity		***Normal = mass velocity***	
		Value	***=1285.7***	*[Kg m^{-2}s^{-1}]*
	Temperature		***= Value***	
		Value	***= 0***	*[C]*
	TurbEnergy		***= Pulsations***	
		Value	***= 0***	
	TurbDissipation		***= Scale of turbulence***	
		Value	***= 0***	*[m]*

  - ***Boundary 4***

Type		**= Blank output**	
Variables			
	Velocity	**= Pressure**	
	Value	**= 0**	**[Pa]**
	Temperature	**= Zero gradient**	
	TurbEnergy	**= Zero gradient**	
	TurbDissipation	**= Zero gradient**	

eserved.

© CAPVIDIA, 1999-2015 Leuven, Belgium. All rights reserved.

1.10 Defining Initial conditions

Initial conditions- this field values calculated variables in a given object initial time.

By default, all variables (except for**the phase** volume)at the initial time are zero. Variable**volume phase**has a default at the initial time value of**1.**

Set the initial conditions only if you want to start calculating with the values that differ from those used by default (zero for all variables, except for**the phase** volume,and a value of**1**for the variable**volume** phase).

Initial conditions are defined in a**Subregion.**

Prior to **theinitial conditions**in the folder properties**subregion #i**must select a predefined**model.**

In order to set the**Initial conditions**

- Set the**Model Initial data:**
  - Create**initial data** [1)]
  - Assign values to create an **initialdata**
- Set the**Property:**
  - Create an**object**for the application of**the initial conditions**in the folder**Objects** [2)]
  - Set the parameters of **the object**
- Set the**Initial** conditions:
  - Create afolder in the**Initial conditions****Initial conditions** in a**Subregion** [3)]
  - Set the parameters of**Initial conditions**

For in this example (mixer)

The initial conditions is not required.

Notes:

[1)]Some**initial data**are always created in the default **model.**

[2)]**Computational space**objectis always created in the folder**Objects**by default.

[3)]Some**Initial** conditions,containing the**initial data # 0**and to**Computational space** are always automatically createdin a**Subregion.**

Assignment of non-zero initial conditions is useful in order to accelerate the convergence of the solution (for example, in the problems of external flow is convenient to define the initial velocity field corresponding to the free stream velocity).

© CAPVIDIA, 1999-2015 Leuven, Belgium. All rights reserved.

1.11 Defining a computational grid

Computational grid- a set of cells into which the computational domain. The values of variables within a single cell constant.

Specifying**the computational grid**includes

- Entering **thestarting grid**
- Specifying**Adaptation**

Specifying the computational grid consists of two stages:

a) specify the start grid. In this example, set the initial uniform computational grid, ie with constant steps of each of the coordinates with a number of steps:

N_X= 20, N_Y= 20, N_Z= 30;

b) job adaptation and adaptation to solution. In this example, set adaptation (local refinement) of the computational grid near the boundaries of the region; adaptation to the solution not to ask.

Defining an initial grid

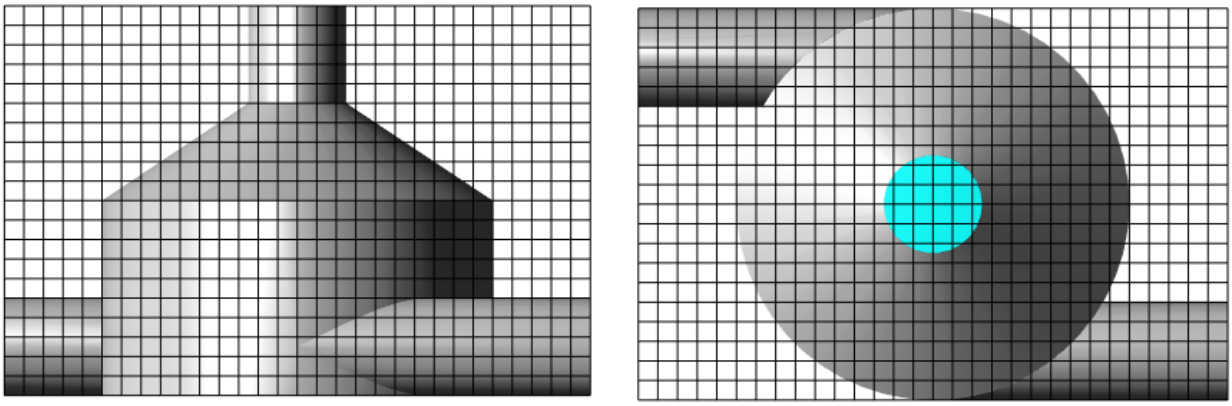
The initial grid is defined in dialog window**Properties of the initial** mesh.

In order to set*uniform* **Starting** grid:

- In the**Properties**window, specify the number**of initial grid**cells in each direction in the respective nX nY nZ

In order to set*an uneven* **Starting** grid:

- In the **Properties** window of the **Initial grid** element, call one of the **Initial grid editors** clicking on or
- In the window that appears, the corresponding **Initial grid editor** set grid options.



The flow in the mixer:

Ask a uniform initial mesh 20h20h30.

- *In the Properties window, set the initial grid:*

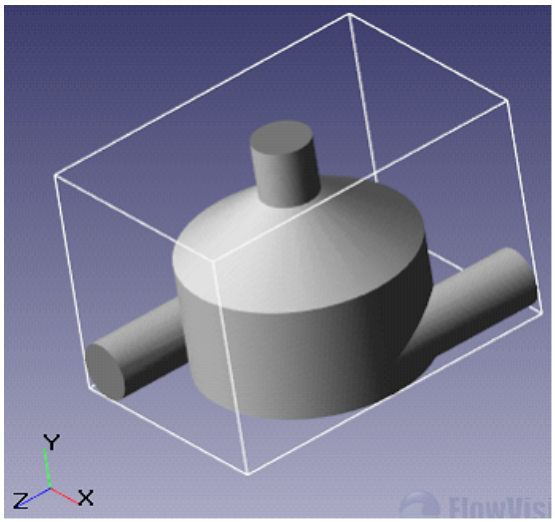
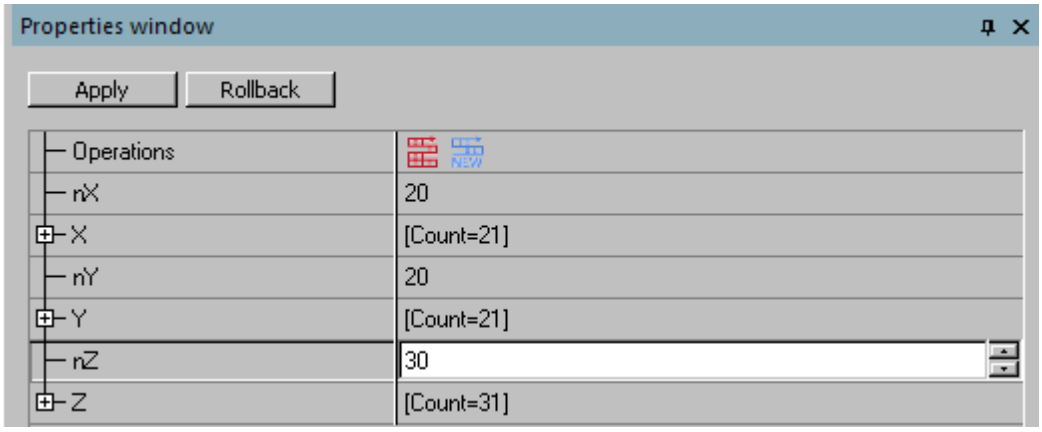
  nX = 20

  nY = 20

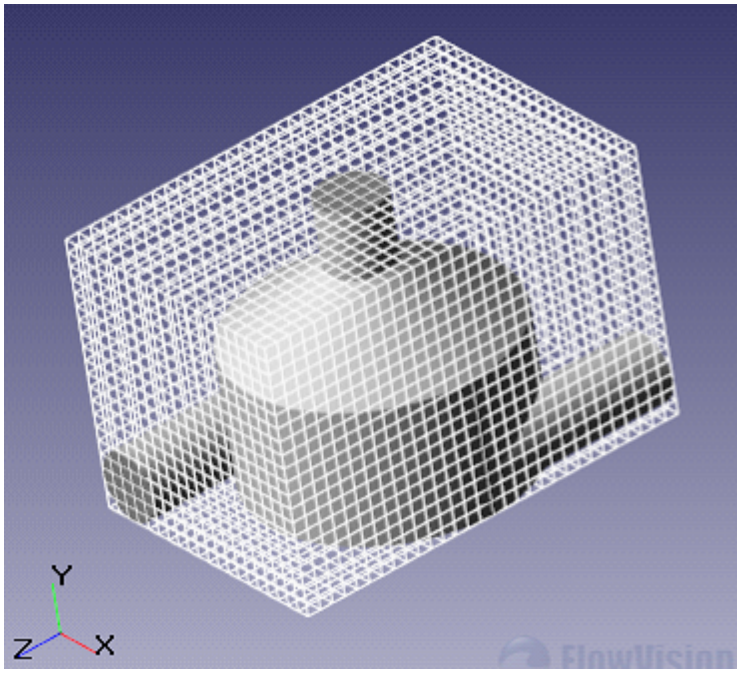
  nZ = 30

Perform the following steps:

© CAPVIDIA, 1999-2015 Leuven, Belgium. All rights reserved.

Step	Actions
1	On the Advanced tab,scroll to the folder**PreprocessorInitial** grid. In**the View**windowshows the computational domain in the "box": 
2	In the**Properties**window,set the number of steps for each coordinate: value <20> for the nX and nY, the value of <30> for nZ, then click**Apply:**  In**the View**windowshows the picture constructed initial computational grid:

© CAPVIDIA, 1999-2015 Leuven, Belgium. All rights reserved.



Specifying adaptation near the borders of the area

Adaptation- a partition of grid cells in each direction in half the specified number of times.

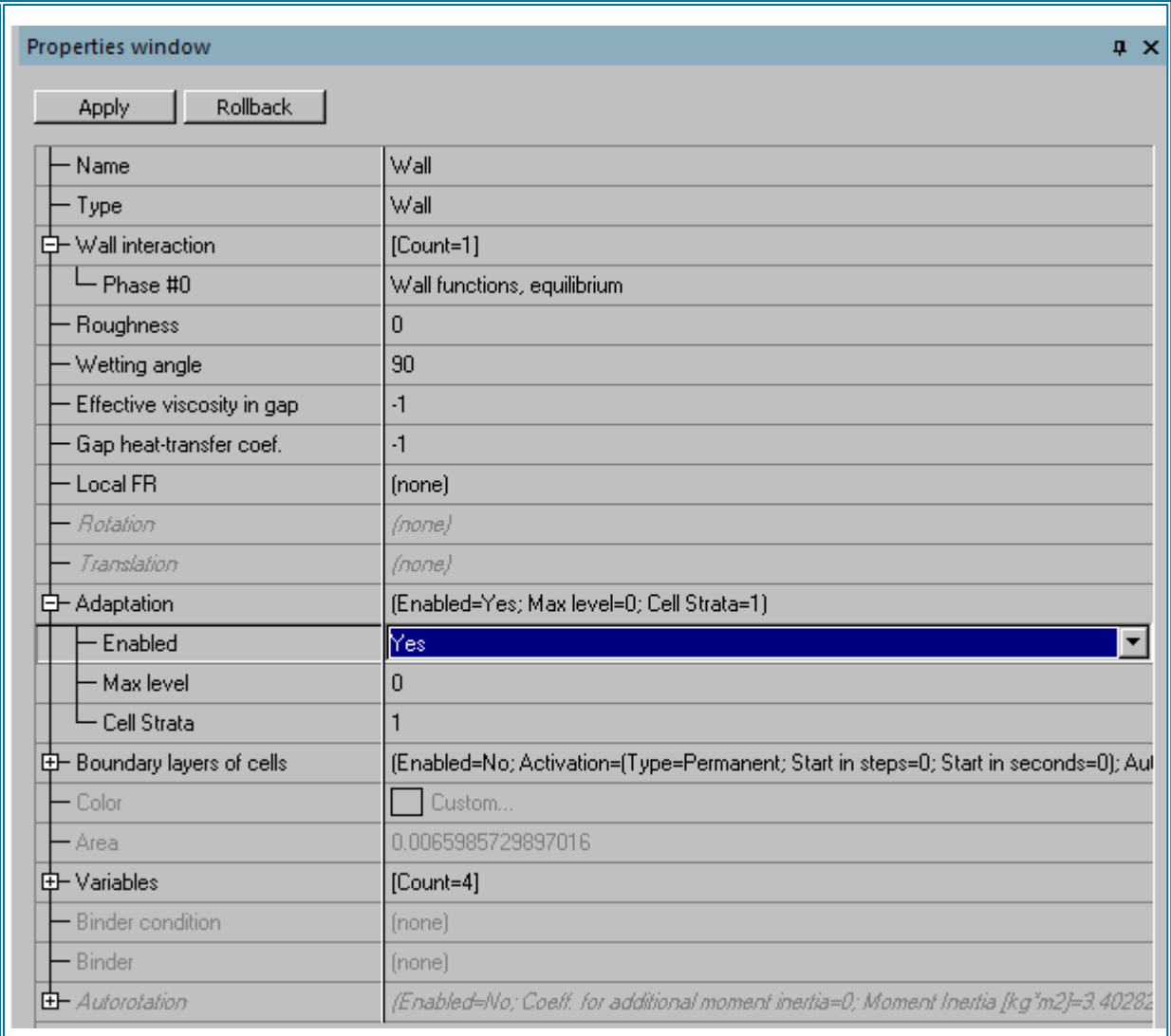
Level of adaptation- the number of consecutive partitions mesh.

Adaptationcan be defined on the**boundary** conditions,or at **thefacility.**

To **be adapted**to the**boundary** conditions,follow these steps:

Step	Actions
1	In the**Properties**window **Boundary conditions**including**adaptation**and set its parameters. For this tab**Preprocessor**project tree select the folder**Subregions > subregion # 0>Boundary conditions>Wall**and its**Properties**window, set the options: a) set**Adaptation> Enable = Yes**(select from the list) b) set**Adaptation> Max. Level = 1** c) set**Adaptation Layers> cells = 2**

© CAPVIDIA, 1999-2015 Leuven, Belgium. All rights reserved.



2	Press **Apply**

The task of adaptation at the Facility

To specify**an Adaptation**to the**object, follow these steps:**

Step	Actions
1	Set the**object**in the folder**Objects**
2	Create**an Adaptation #N** folder **Adaptation** corresponding**Subregion**
3	Set the parameters**Adaptations**

© CAPVIDIA, 1999-2015 Leuven, Belgium. All rights reserved.

For in this example (mixer)

*Set adaptation level 1 near the surface on which the identified boundary condition**Wall:***

- *In the properties of the boundary conditionsset **Wall:***

Adaptation	Yes
Level	1
Cell Layers	2

© CAPVIDIA, 1999-2015 Leuven, Belgium. All rights reserved.

1.12 Defining simulation controls

Setting calculation is carried out on thetab **Solver.**

The main control parameters calculation includes the step of integration of the model equations in time and stop conditions calculation.

Specifying the time step

The time step is set in the**Properties**window Time step.

The time step can be set:

- permanent
- by the Courant-Friedrichs number-Levy

In order to set the time step, in the**Properties**window**of the time** step:

- specify the way in which will be determined by the time step
- then set the values needed to determine the time step chosen method

Constant step is recommended to set equal to 1/10 of the characteristic time. In the problem of flow in the pipe as a typical time span can be selected - that it would take a particle released from the entrance of the computational domain in order to reach the exit of the computational domain. And the problem of flow around an aircraft as a characteristic linear dimension can take the size of the device in the direction of flow, and as a characteristic velocity - the speed of the incoming flow at infinity.

The flow in the mixer:

*Ask a**permanent step = 0.001**s.*

Setting conservation

The results of the calculation can be saved automatically in the process of calculating the specified interval. If necessary, it is possible to preserve the history of calculation. You can store the data and layers. Saving data is configured in the Autosave data. Saving layers can be adjusted in the AutoSave layers.

The operations with saved results inPre-Postprocessor:

Operation		Data	Layers
Running on the computation of a stored step		Yes	No
Visualization in touch with**Colverom**		Yes	No
Visualization without regard to**Colverom**		No	Yes
Visualization of layers created prior to the calculation of			
	Beds of the parameters did not change after saving	Yes	Yes
	Changed settings affecting the construction**Cloya**	Yes	No[1)]
	Changed settings affecting the display**Cloya**	Yes	Yes
Visualization of layers created after calculating		Yes	No
Automatic sequential loading results in touch with**Colverom**		Yes	No

The operations with saved resultsin**Viewer:**

Operation	Data	Layers
Visualization in touch with**Colverom**	Yes	No

© CAPVIDIA, 1999-2015 Leuven, Belgium. All rights reserved.

Visualization without regard to**Colverom**	No	Yes
Visualization of layers created after calculating	Yes	No
Automatic sequential loading results without regard to the solver	No	Yes

The flow in the mixer:

*Ask**Data autosave**with parameters:*

History		***Yes***
Frequency		
	Type	***By the time***
	The number of seconds	***0.005***

Notes:

[1)]Visualization is performed using the build settings defined before saving.

Specifying Conditions stop

The calculation can be stopped manually by the user through buttons in the toolbar or automatically in accordance with the terms of the stop.

Break conditions:

1. The length of time - Payment stops at the specified time.
2. Iteration - Payment stops when the specified number of iterations
3. Residuals - Payment stops when all the created residuals calculated variables are below the specified values
4. Custom values - Payment stops when all the residuals by custom variables are below the specified level.

Plot residuals calculated variables and values of user variables is displayed in the**Monitor**tab**Plot.**

The flow in the mixer:

*Set the**length of** time:*

Starting	**0**
Stop	**1**

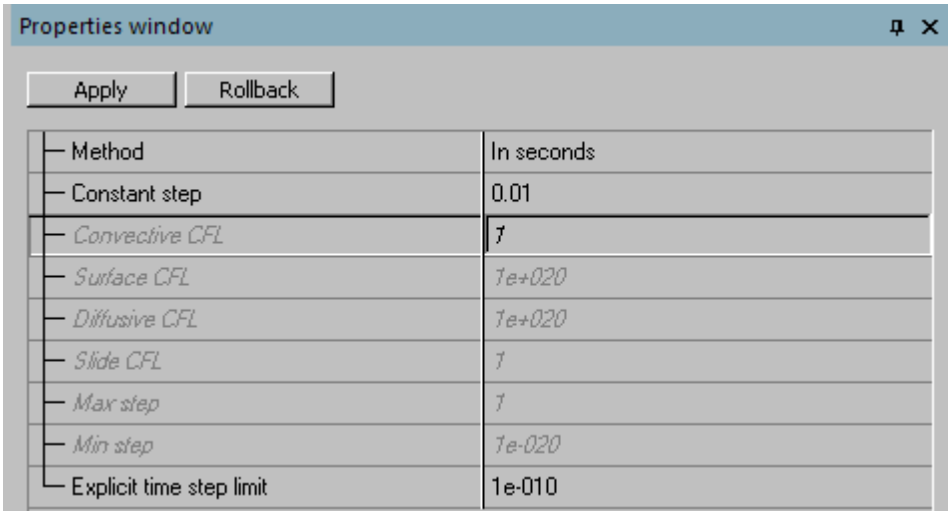
Note:

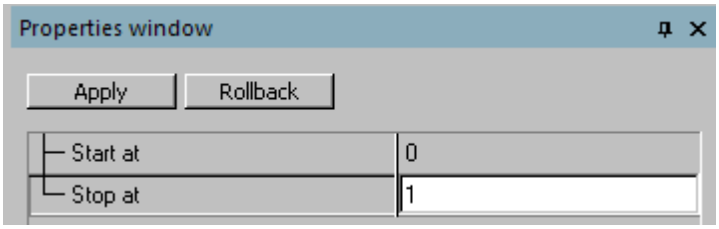
Specifying Conditions shutdown is not necessary for the calculation.

Setting the control calculation for this example

Step	Actions
1	**Solver**tab, scroll to the**time** step,and in the**Properties**window, set the options: a) Select**a**value**in seconds**for the**process**parameters b) set the value of**0.01**for the**constant**parameter**step**

© CAPVIDIA, 1999-2015 Leuven, Belgium. All rights reserved.



2	Click **Apply.**
3	On the Advanced tab, scroll to the **Solver** **Break conditions** > **The length of time** and in the **Properties** window, set the value of parameter **1** **Stop:** 
4	Click **Apply.**

© CAPVIDIA, 1999-2015 Leuven, Belgium. All rights reserved.

1.13 Displaying results

Displaying calculation results (visualization) is provided in**Postprocessor**using **Characteristics** and **Layers**. The conditions under which the mapping may depend on the manner in which the preservation of the results was carried out (in the form of data or in the form of layers).

If carried storing data to display the results obtained, it is necessary:

- open a project in**Pre-Postprocessor**
- connect to**Solver**
- if necessary, to synchronize the client and server parts.

After that, the**Pre-Postprocessor**will display the last saved data. If necessary, you can create additional elements Postprocessor.

If the data is saved to the history, you can view intermediate results (through the toolbar **Navigation) and** creating animations (By the toolbar **Pictures) with the mapping of the dynamics solutions in time.**

Elements to display the results should be chosen depending on the type of the display data:

Data Type	Name of the element
Integral values:	
Variable value on the surface and in the bulk	**Characteristics**
Local values:	
Local values of the scalar variable along the line	**Layer «Plot along line»** **Layer «Plot along curve»** **Layer «Plot along ellipse»**
Local values of the scalar variable at the surface	**Layer «Color contours»**
Local values of the scalar variable in the amount of	**Isosurface** **VOF** (Only for variable VoF, see section Theory> Physical Processes> Transfer phase)
Local value and direction vector variable on the surface and in the bulk	**Layer «Vectors»** **Layer «Streamlines» and element «Emitter»**

Displaying various values:

The value of	Object	Feature / Layer	Variable in the Properties window	Variable in the Data window
Integral values:				
The flow through the input or output	**Supergroup**Created on the surface of the input or output	**Characteristics**in the corresponding **Supergroup**	Any list	Mass flow + Mass flow- Volumetric Flow + Volumetric flow-
The average speed in the section	**Plane** section	**Characteristics**on the corresponding**planes**	**Velocity**	<f surf.>
Average temperature section	**Plane** section	**Characteristics**on the corresponding**planes**	**Temperature**	<f surf.>
The average temperature of the flow through the section	**Plane** section	**Characteristics**on the corresponding**planes**	**Temperature**	<f mass+> <f mass->
The average surface temperature	**Supergroup**Created on a given surface	**Characteristics**in the corresponding	**Temperature**	<f surf.>

© CAPVIDIA, 1999-2015 Leuven, Belgium. All rights reserved.

		Supergroup		
Force or moment acting on a streamlined body	**Supergroup**Created on the surface of the body	**Characteristics**in the corresponding **Supergroup**	Any list	- F liq. X, Y, Z[1)] - M liq. X, Y, Z[1,2)]
Position of the center **Moving body**	**Imported object**, On which a **Moving body** is created	**Characteristics**on the corresponding**import objects**	Any list	The center of rotation X, Y, Z
Velocity of movement **Moving body**	**Imported object**, On which a **Moving body** is created	**Characteristics**on the corresponding**import objects**	Any list	Velocity X, Y, Z
Local values:				
Plot of the pressure distribution along the contour of the cross section	**Plane** section	**Plot along curve** corresponding to **Plane** [3)]	**Pressure**	
The pressure distribution in the cross section	**Plane** section	**Color contours** corresponding to **Plane**	**Pressure**	No
The velocity distribution in the cross section	**Plane** section	**Vectors** corresponding to **Plane**	**Velocity**	No
The pressure distribution on the body surface	**Supergroup**Created on the surface of the body	**Color contours** corresponding to **Supergroup**	**Pressure**	No
Streamlines	**Computational space**	**Streamlines** in **Space** [4)]	**Velocity**	No
Surface phase (only in problems with the contact surfaces)	**Computational space**	**VOF**	No	No
Pressure in a given volume (sensor)	**Object** (**Box** or **Ellipsoid / sphere** or **Cone / cylinder**) corresponding sensor	**Characteristics to facilities**	**Pressure**	<f об.>

Create a layer and change its settings can be performed at any time

When conducting long-term calculations should always display the data in the calculation process, as in this case, the user is able to permanent control over the process of convergence of the solution and, if necessary, to intervene in the process of calculating when a numerical instabilities or divergence of solutions. Sometimes the user makes a mistake when setting the option and the majority of these errors are detected in the first iteration of the computational algorithm by observing the behavior of the solution.

Notes:

[1)]$F_{liq.}$ and $M_{liq.}$, respectively, the force and the moment that the body acts on the flow. Force and the moment that the flow acts on the body, are equal in magnitude and opposite in sign $F_{liq.}$ and $M_{liq.}$.

[2)]The moment is defined relative to the **center**,whose coordinates are given in the **Characteristics.**

[3)]When you create a **Plot along curve** requires that the curve was in the first quarter (quadrant) of the coordinate system graph.

© CAPVIDIA, 1999-2015 Leuven, Belgium. All rights reserved.

[4)] On **Streamlines** also necessary to set the **source. As a source can be any object.**

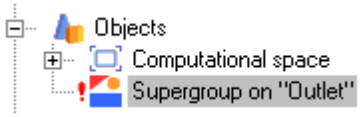
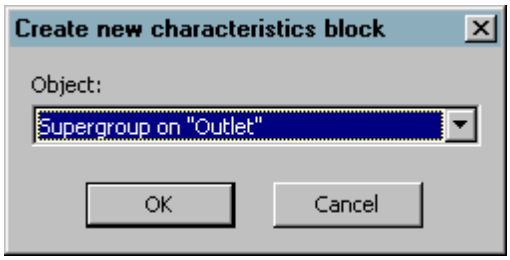
© CAPVIDIA, 1999-2015 Leuven, Belgium. All rights reserved.

1.13.1 Creating charachteristics

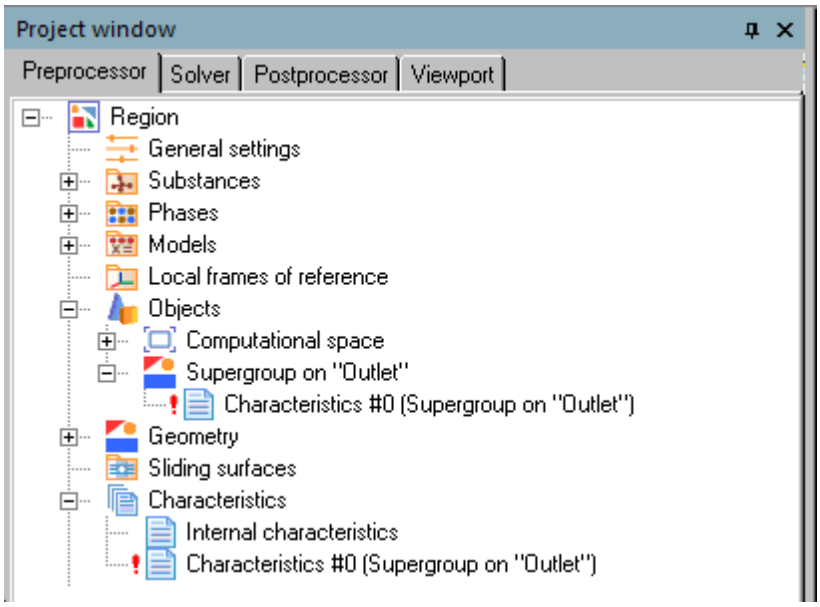
To control the output flow regime a steady state will monitor the temperature at the exit of the mixer. To do this, follow these steps:

Step	Actions
1	First you need to select a computation Characteristics - A set of integral values (including secondary) obtained after integration of a variable on the surface or in the bulk of the object. Job characteristics is reduced to the direction of the variable and the object on the surface or volume which should be integration. To do this, select the output section to set up an a **Supergroup**- element of the project tree containing facets, which supplied the output boundary condition.
	We use the fact that the component characteristics included in the calculation of the number of stopping criteria is displayed in the form of a graph in the **Monitor,** and define stopping condition calculation the criterion on the basis of characteristics of components to the average value of the variable.

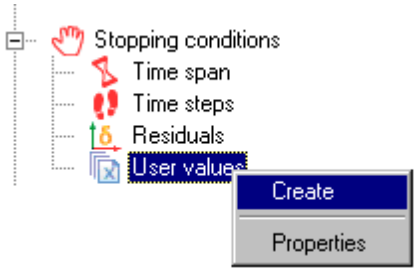
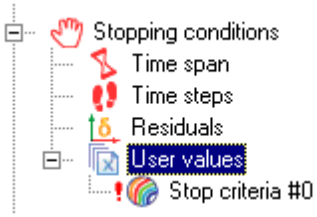
To create Characteristicson a Supergroup:

Step	Actions
1	Open the tabcontext menu item**PreprocessorSubregions> subregion # 0>Boundary conditions>Exit**, and then click **Create supergroup>preprocessor.** In the project tree is added created supergroup (in our example - an element **Supergroup on "Outlet"**in the folder**Objects):** 
2	Open the context menu on the element **Supergroup on "Outlet"** and select **Create** characteristics. A dialog box **Create new characteristics block**: 
3	Click**OK.**

© CAPVIDIA, 1999-2015 Leuven, Belgium. All rights reserved.

	In the project tree element is added **Characteristics # 0 (Supergroup on "Outlet")**: 
4	Select the element **Characteristics # 0**, in the **Properties** window, select the value for the parameter **High Characteristics > Variable** and click **Apply.**

To set the stop condition for calculating the criterion on the basis of characteristics:

Step	Actions
1	Open the tab **Solver** context menu item **Break conditions > Custom values** and select the command **Create:**  In the project tree is added created a stopping criterion (in our example - an element of **a stopping criterion # 0** in the folder **Break conditions > Custom** value): 
2	Select the item **stopping criterion #** 0, in the **Properties** window, do the following: a) select **Characteristics # 0** for the parameter **object** b) select **<fweight->**[*)] for the parameter **variable** (thereby setting the calculation of average cross section of the outlet temperature of the water *flowing* through the outlet) c) select **a** value of **0** for the parameter **level** (that is, computation will be interrupted, unless the characteristic will be constant over time) And click **Apply.**

© CAPVIDIA, 1999-2015 Leuven, Belgium. All rights reserved.

	*)If you select **<fweight +>**,it would mean computation for the water *flowing*through the outlet that would have led to a null result.

For in this example (mixer)

*Create **Characteristics** for **Supergroup**, the corresponding boundary conditions **outlet**:*

- *Create based on the boundary conditions **outlet a Supergroup** in **Postprocessor***
- *Create **Characteristics** for **Supergroup***
- *In the **Properties** window **Characteristics**set*

Characteristics			
	Variable		
		Variable	**Temperature**

- *In order to maintain the momentum of change **of temperature** in the file, before the start of the calculation set:*

Save to File		
	Mode	**Automatic**

1.13.2 Creating layers

For visual inspection of the calculation will follow the velocity vectors and temperature in the two sections of the mixer. To do this, follow these steps:

Step	Actions
1	First you need to create two planes: • one of them - the vertical plane of symmetry of the mixer • second - the horizontal plane passing through the axis of lead pipes
2	Further, these planes define layers that define the mapping in a color temperature of the fill, and the speed - in form of vectors. Each layer displays a certain way only one variable and only one object. It is therefore necessary to create two layers on the same plane, and two - on the other. In one layer couples the temperature field to be displayed in the other - the velocity vector.

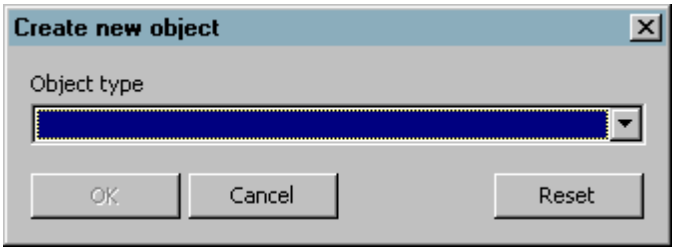
In order to set the**layer,**

- Create a**preprocessor** or**Postprocessor Object**and set its properties
- Create a**layer**to **the created object**

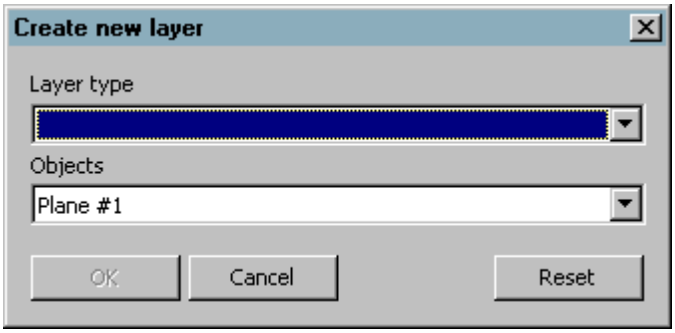
To create planes:

Step	Actions
1	Open the tabcontext menu item**PostprocessorObjects**and select the command**Create. A dialog box Create new** object:

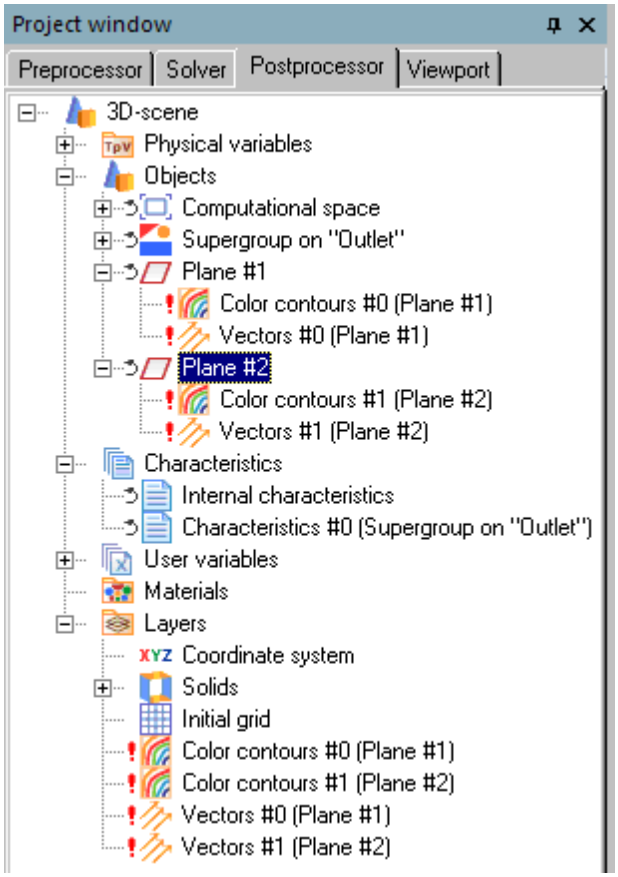
© CAPVIDIA, 1999-2015 Leuven, Belgium. All rights reserved.

	
2	Select the **Object type** field value **plane, and then click OK. In the project tree is added to create a plane -Plane # 1** item in the folder **Objects, which coincides with the desired vertical plane of symmetry of the mixer.**
3	Open the context menu on the element **plane # 1** in **Objects** folder and select **Copy. In the project tree, the plane -plane** element **# 2** in the folder **Objects.**
4	Scroll to the element **plane # 2** and in the **Properties** window, do the following: a) click **Object> Operations>** and click **Apply; plane will be horizontal;** b) select **0.005** for the parameter **object> reference point>** Y, and then click **Apply; horizontal plane passes through the axis of the underwater tubes.**

To set the layers:

Step	Actions
1	Open the context menu item **Objects> Plane # 1** in tab **Postprocessor** and select the **CreateLayer. Dialog box opens Create new** layer: 
2	Select **Color contours** for the parameter **type layer** and click **OK.**
3	Open the context menu item **Objects> Plane # 1** and select the **CreateLayer.**
4	In the dialog box, select the value for the parameter **vectorstype layer** and click **OK.**
5	Repeat steps **1**-**4** for the element **Objects> Plane #** 2. In the project tree displays created four layers:

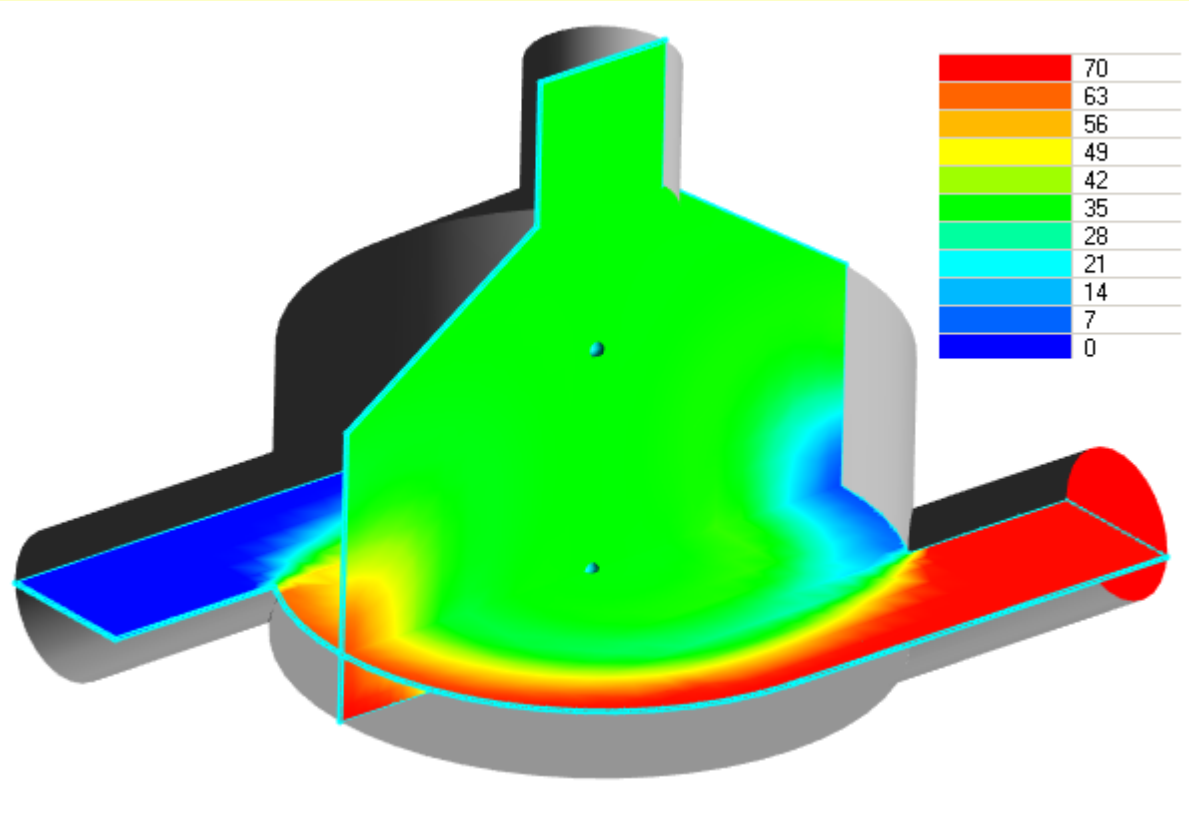
© CAPVIDIA, 1999-2015 Leuven, Belgium. All rights reserved.

	
6	Set the parameters of both layers **Color** contours, highlighting an item in the project tree, and performing the following steps in the **Properties** window: a) select **temperature** for **Variable>Variable** b) select **Manual** for setting **range> Mode** c) set the values of **70** and **0** for the parameter **range> High** and **Range> Minimum**, respectively and click **Apply.**
7	Set the parameters of both layers **vectors, selecting an item in the project tree, and performing the following steps in the Properties** window: a) select **Velocity** for **Variable>Variable** b) specify a value of **21** for the parameters **Grid>GridSize 1** and> 2 Size c) select **Manual** Scaling> Mode and click **Apply.**

The layers are defined in the folder **Layers** in **Postprocessor** on the basis of **pre-created** object.

© CAPVIDIA, 1999-2015 Leuven, Belgium. All rights reserved.

For in this example (mixer)



The flow in the mixer:

Ask two layers ***Color contours*** *for the variable* ***temperature*** *- one on the plane crossing the inlets, and the other - on the plane of symmetry of the mixer:*

- *Create two* ***planes;***
- *In the* ***Properties window*** *of the first plane is given by:*
  - ***Objects> Reference point> X = 0***
  - ***Objects> Reference point> Y = 0.005***
  - ***Objects> Reference point> Z = 0***
  - ***Objects> Normal> X = 0***
  - ***Objects> Normal> Y = 1***
  - ***Objects> Normal> Z = 0***
- *In the* ***Properties window*** *the second plane is given by:*
  - ***Objects> Reference point> X = 0***
  - ***Objects> Reference point> Y = 0.02***
  - ***Objects> Reference point> Z = 0***
  - ***Objects> Normal> X = 1***
  - ***Objects> Normal> Y = 0***
  - ***Objects> Normal> Z = 0***
- *Create Layers of* ***Color contours*** *on both* ***planes***
- *In the* ***Properties*** *window, set of* ***Layers: Variable> Variable = Temperature***
- *In order to display the* ***temperature*** *in the same scale on both layers, specify:*
  - ***Range> Mode = Manual***
  - ***Range> Maximum = 70***
  - ***Range>= 0 Maximum***

ved.

1.14 Carrying out the project calculations

Details on the calculation of the project start, stop and continue the calculationsee in section Start, stop, and continue the calculation of project.

The following describes the launch of the calculation of this example.

For in this example (mixer)

*Run the project on the computation of **pre-postprocessor:***

- *Log on**Solver-Agent:***
  - *Press* in the toolbar
  - *In the next window, enter the user name and password on the**Solver-Agent***
- *Project to the solver*
  - *Press* in the toolbar
  - *In the window that appears, set the parameters of the**Solver***
  - *Run**Solver***
  - *Highlight running**Solver**and click**Connect***
- *Run**Solver**in computation*
  - *Press* in the toolbar
  - *In the window that appears**Starting solve**remove all tags*
- *If necessary, stop computation via buttons* in the toolbar

Authorization for the**Solver**possible only when the **Solver-Agent**. Before operating the**Solver**run**Solver-Agent.**

© CAPVIDIA, 1999-2015 Leuven, Belgium. All rights reserved.

1.15 Monitoring the project calculation process

The main methods of control in the process of calculation of the project are as follows:

- basic control parameter calculation (including graphics predetermined stop condition calculation) in the **Monitor**
- imaging layers constructed in the course of computation;
- control characteristics in the information window**Info.**

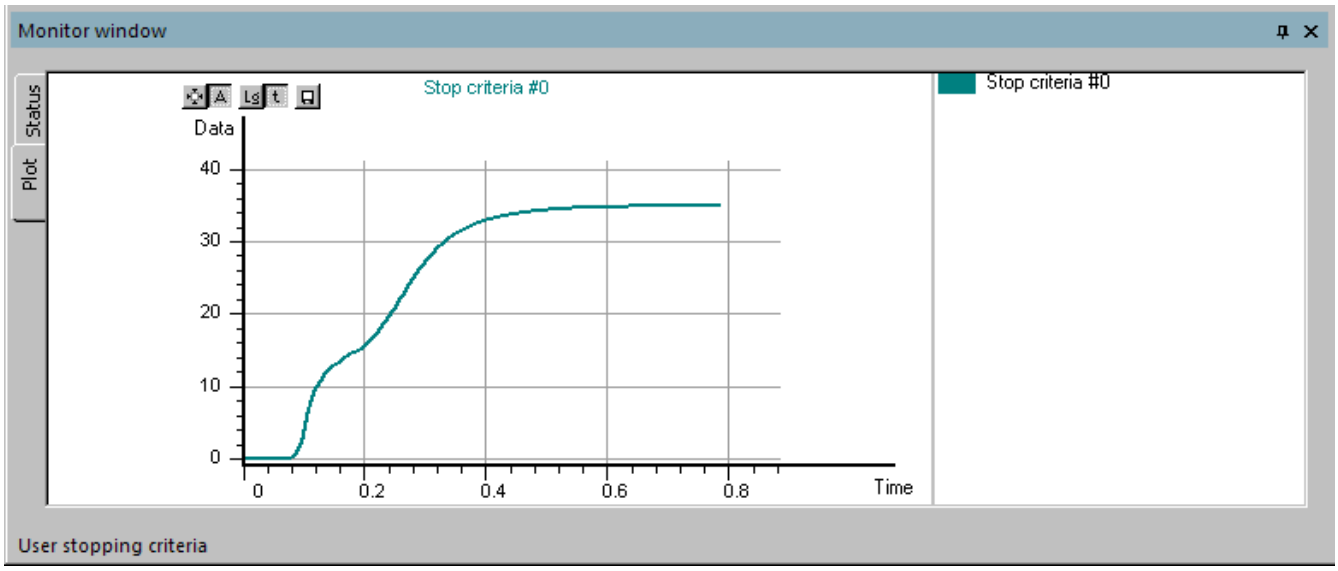
Control of the main parameters of the calculation in the «Monitor»

Select the display mode of the**Monitor**windowand follow in the course of the calculation:

a) **Status**modeto track changes of the following parameters:

- number of iterations, which should not significantly increase (strong growth means instability account)
- Algebraic residuals that must be within a predetermined accuracy of calculation
- functional residuals, which should eventually decrease to a low value corresponding to steady flow, or oscillate with a limited amplitude

b) **Plot**modeto track the changes of the control parameter over time (see. figure).



Displaying a dependency ofthe control parameter from time to time in the «Monitor»

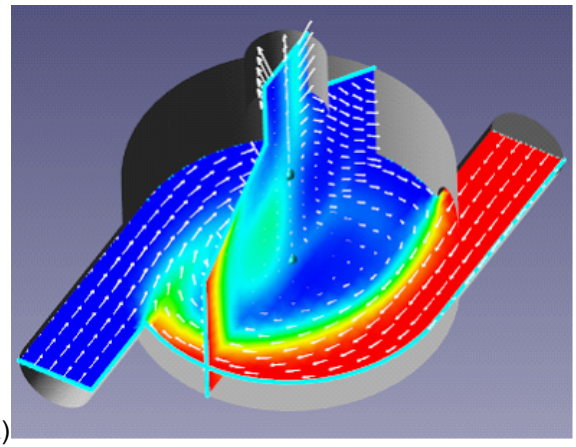
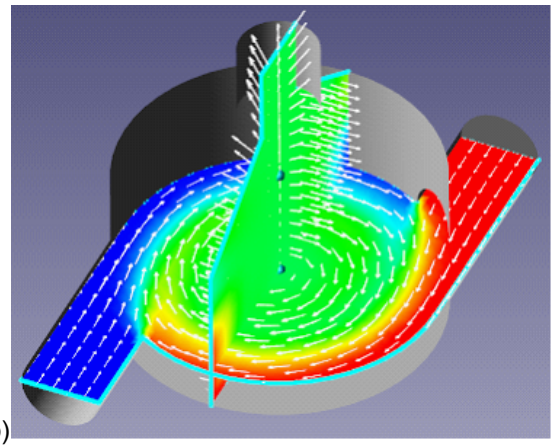
Displacement or stretching of the scale of the axes graph:

Step	Actions
1	Move the mouse pointer to the line axis of the graph.
2	After changing the shape of the pointer, press the left mouse button and keep it pressed, move the mouse pointer along the axis.
3	Release the left mouse button.

Visual inspection of the project calculation

1. Press (Turn on / off the external surface)in the toolbar **Solids, to make transparent object boundaries, which enclose the interior space. This allows you to see the layers on the intersecting surfaces (see. Figure).**

© CAPVIDIA, 1999-2015 Leuven, Belgium. All rights reserved.

a)  b) 

Displaying a field of temperature and vectors of velocity on layers:
and - an intermediate time point b - final result

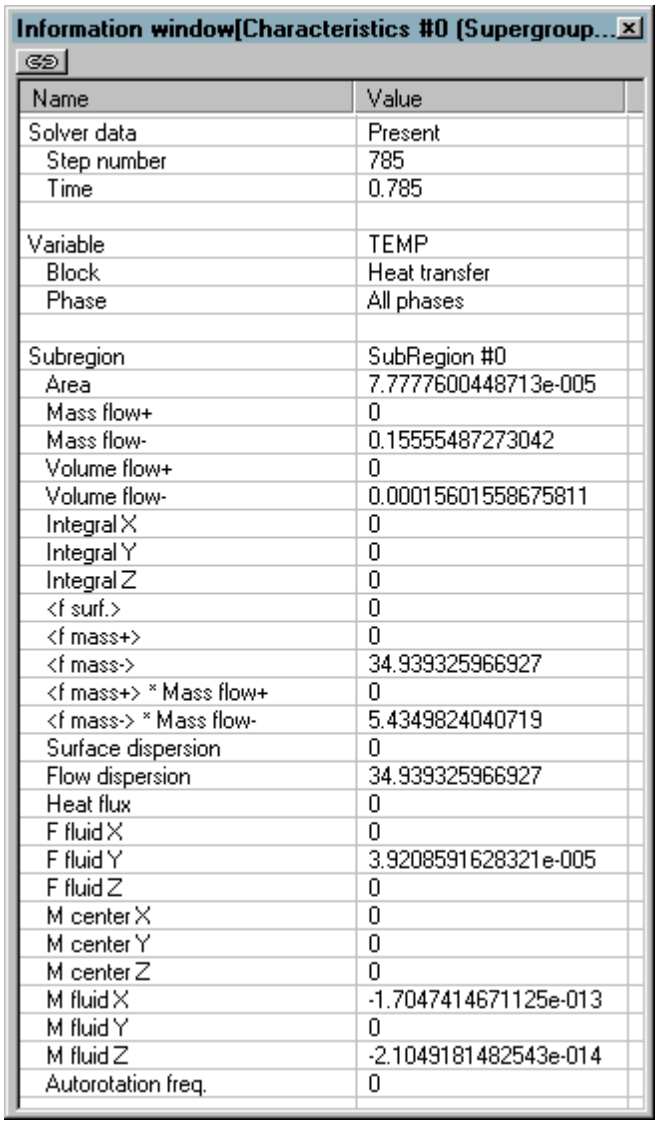
2. Press (View transformation mode)in the toolbar **Modes** to select the conversion type and choose the most convenient angle for the analysis of the flow.

Control characteristics in the information box «Info»

Highlight the project tree line characteristics, and then click (See the information box)in the toolbar **Modes.**

Window opens**Info**(Fig. 3.5) with the parameters characteristics(para 7.6.1),calculated on the last time step (their values are updated at the end of the next time step).

© CAPVIDIA, 1999-2015 Leuven, Belgium. All rights reserved.



Name	Value
Solver data	Present
Step number	785
Time	0.785
Variable	TEMP
Block	Heat transfer
Phase	All phases
Subregion	SubRegion #0
Area	7.7777600448713e-005
Mass flow+	0
Mass flow-	0.15555487273042
Volume flow+	0
Volume flow-	0.00015601558675811
Integral X	0
Integral Y	0
Integral Z	0
<f surf.>	0
<f mass+>	0
<f mass->	34.939325966927
<f mass+> * Mass flow+	0
<f mass-> * Mass flow-	5.4349824040719
Surface dispersion	0
Flow dispersion	34.939325966927
Heat flux	0
F fluid X	0
F fluid Y	3.9208591628321e-005
F fluid Z	0
M center X	0
M center Y	0
M center Z	0
M fluid X	-1.7047414671125e-013
M fluid Y	0
M fluid Z	-2.1049181482543e-014
Autorotation freq.	0

Fig. 3.5. Information window «Info»with parameters "Features # 0"

© CAPVIDIA, 1999-2015 Leuven, Belgium. All rights reserved.

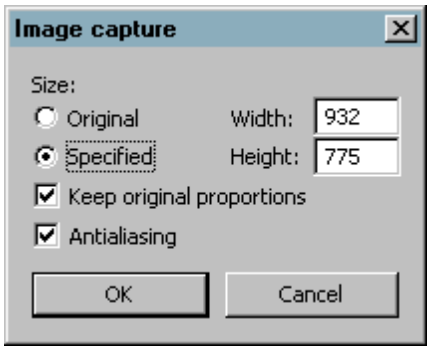
1.16 Creating an animation

Animation is created from a set of graphic files created either during the account or when viewing saved data. These image files contain the image of the window **View.**

Here is how to create animation in the**Pre-Postprocessor**during counting.

Note:Similar opportunities are also available in animation **Viewer**.

Creating animation during counting

Step	Actions
1	Set the layers, the image which you want to save. On the layers is recommended to set**Range = Manual**to the range and, accordingly, the color of layers depending on the displayed variable values did not change from step to step.
2	Press (Start Capture sequence **Begin** sequence)in the toolbar **Pictures.**Dialog box opens**Image** Capture: 
3	Ask in the**Image Capture**settings (size, elimination of aliasing). Click**OK.** (For a detailed description of the elements of this window, see. Section Management of capturing images).
4	This opens the standard Windows operating system to access the files **(SaveImage).** Specify the location, type, and name prefix for the files that will be written to the image. Then click the**Save**button. Images will be written to the file at each updatewindow**View.** The file names are formed as follows: **(Prefix)** _00000. (Filetype) **(Prefix)** _00001. (Filetype) **(Prefix)** _00002. (Filetype) etc.
5	Create a video from a sequence of image files saved with the help of a specialized program.

The flow in the mixer:

*Create animation in the process of changing the account **of thetemperature**distributionin the planes described in the example in section Creating layers.*

© CAPVIDIA, 1999-2015 Leuven, Belgium. All rights reserved.